Course of "Fluid Labs"

Some tips for PHOENICS

Dr. Ing. Luca Nicola Quaroni

Installation of PHOENICS 2022 on Windows 46 bit computers

Useful sources of information

Reference page for PHOENICS (here) can be a little «criptic» sometimes.

Here are some useful references:

- Phoenics-VR reference guide (here) and (here)
- Convergence monitoring (here)
- Turbulence models (here)
- General tutorials (here)

Tips on various topics

Suggest inserting folder *d_priv1* to favorite folders.

Click on the right button and roll the mouse roller: much faster zoom-out.

Applying cylindrical coordinates x becomes θ (in radians), y becomes r and z is unchanged.

A *plate* object has no thickness in one direction, for the pipe select the 'y plane'. In the size dialog, toggle 'to end' for both x and z. In the 'place' dialog window, toggle 'to end' of y and leave x and z to zero.

The Q1 file is the "instructions" file to modify if you want other functionalities.

To save intermediate phi files go to 'output', 'field dumping' and toggle on 'intermediate phi dumps'. Then select the sweep frequency. These can be imported through the same MATLAB scripts.

A useful tutorial is given here

Useful tutorials and worked-out examples can be found by going in 'file', 'load from library' and selecting a number.

To save a quantity and its time history go to 'settings', 'domain attributes', 'inform', 'create new save block'. You'll have a selection of 25 new places where to store commands in InForm language (click here for more information). To store the residuals in a residual.csv file in d_priv1:

```
save13begin
(STORED OF PRES IS RESI(P1))
(STORED OF WRES IS RESI(W1))
(STORED OF VRES IS RESI(V1))
(TABLE in residuals.csv is GET(PRES,VRES,WRES) with HEAD(PRES,V1RES,W1RES)!SWEEP)
save13end
```

Remember that in a laminar flow simulation, the wall shear stress is computed by subdividing the velocity at the centroid with the distance of the centroid from the wall. Hence, no restrictions on the dimensions of the first layer of cells are required, even though in order to approximate the flow it's necessary to follow the steep velocity gradient nonetheless. Otherwise, the gradient is not accurately predicted!

Remember that in the case of cylindrical coordinates, the axis condition is applied automatically.

An interesting discussion around convergence criteria is given here.

Some reference on mesh generation is reported here.

Notes on convergence monitoring

There are four main tools that are used to determine whether reasonable convergence has been achieved:

- source imbalances
- sums of absolute residual errors
- spot value behaviour
- maximum absolute corrections

<u>Source imbalances</u>

Good source balance should give a discrepancy between the positive and negative sums that is a small percentage of either; values of <1% should usually be achieved, and considerably lower figures are not uncommon - but a law of diminishing returns applies, with the cost in CPU time of the extra convergence being rather greater than that required to get to a reasonable level. Of course, not all variables can be expected to balance: the effect of solid obstructions enters the momentum equations through the pressure gradient, which is a built-in source rather than an externally imposed one - it does not therefore appear in the source balance. Typically, mass (R1/R2) and energy will balance, as will most other scalars, but care is still needed: if the value of a scalar is given a fixed value anywhere, the source required to preserve that value is NOT included in the source print-out. Source imbalance is a clear indication that convergence has not yet been achieved; source balance does not, though, necessarily indicate that convergence has been achieved - the behaviour of the residuals, spot-value and variable maximum corrections should also be considered.

<u>Sums of absolute residual errors</u>

Residuals are the imbalances (or errors) in the equations for each solved variable, summed over the cells within the computational domain. These can either be absolute (SELREF=F) or relative (SELREF=T); in the latter case the imbalances are scaled with respect to a value that is intended to represent the flux of the variable throughout the domain. Interpretation of residuals can be difficult and very subjective. What is certainly true is that residual values should typically go down by at least a factor of 100 from the value after the initial few sweeps (assuming that the calculation is starting from an arbitrary initial state). Eventually, the residuals are likely to level out, with small oscillations about a fairly constant value. This is usually an indication of convergence, but not always: too tight relaxation can sometimes suggest this sort of behaviour because variables are not able to change by much on each sweep, while too loose relaxation can prevent residuals falling further because the variable values are oscillating.

<u>Spot value behaviour</u>

Spot value behaviour is therefore useful in determining whether or not the levelled residuals can be trusted! If the spot values in a representative region of the flow have settled down to a more-or-less constant value, it is reasonable to assume (if the residual behaviour looks promising) that convergence has been achieved; if the changes are still significant, convergence has not been achieved. Some care is still needed though: apparent settling down of spot values might be caused by too-tight relaxation, resulting in a very slow drift that can be mistaken for real convergence.

Maximum absolute corrections

Apart from residual errors, it is possible to monitor the absolute values of the largest corrections anywhere in the domain. Once the largest correction falls to zero, or at least a negligible fraction of the value being corrected, it would be reasonable to assume that convergence has been achieved, even if the sum of the residuals has not fallen below any specific level. You can create a plot of the maximum absolute corrections from VR, by selecting: Menu / Output / Monitor graph style / Max abs cor. This can be a very useful measure of convergence

Notes on transient analysis (adapted from here)

Fully implicit backward differencing is employed for the transient terms, and central differencing is used for the diffusion terms. The convection terms are discretised using hybrid differencing in which the convective terms are approximated by central differences if the cell face Peclet number is less than 2 and otherwise by upwind differencing.

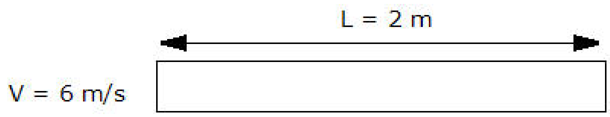
It may be useful to initialize a transient simulation by importing the corresponding steady-state flow field.

The history of selected variables during the iterations in a steady-state simulation can be saved in a .csv file by inserting the following In-Form function:

```
  save13begin
(TABLE in test.csv is GET(V1[63,40,1]) with HEAD(V1)!SWEEP)
  save13end
```

A good rule of a thumb for the choice of a first-attempt time step in transient simulations can be found here.

For transient flow solutions, it is important to select an appropriate time step size. A time step that is too large will result in lost detail because it exceeds the time scale of the flow. A time step that is too small will capture the flow detail, but will not be efficient because it requires more time steps than necessary to characterize the time scale of the flow.



A good guideline for the time step size is approximately 1/20th the time required for a particle of fluid to traverse the length of the device. For example, liquid travels 6 m/s through a 2 meter pipe. It takes 0.33 s for a particle to traverse the length of the pipe. Following our guideline above of 1/20th the time, use a time step of 0.0167 seconds:

- *Total Travel time = L/V = 2m / 6m/s = 0.33s*
- *Time step size = 0.33s x (1/20) = 0.0167s*

Pay attention to the starting letter of dumped files: if left to “null” in the unsteady simulation, the Z-axis is time in the .parphi file. If a letter is added, it will import it as if “dumped” files for a sweep (iteration).